

## BRUSH SEAL NUMERICAL SIMULATION: CONCEPTS AND ADVANCES

M.J. Braun and V.V. Kudriavtsev  
Department of Mechanical Engineering  
University of Akron  
Akron, Ohio

23079  
p. 39  
07183  
322152

The development of the brush seal is considered to be most promising amongst the advanced type seals that are presently in use in the high speed turbomachinery. The brush is usually mounted on the stationary portions of the engine and has direct contact with the rotating element, in the process of limiting the "unwanted" leakage flows between stages, or various engine cavities. This type of sealing technology is providing high (in comparison with conventional seals) pressure drops due mainly to the high packing density (around 100 bristles/ $1 \text{ mm}^2$ ), and brush compliance with the rotor motions. In the design of modern aerospace turbomachinery leakage flows between the stages must be minimal, thus contributing to the higher efficiency of the engine. Use of the brush seal instead of the labyrinth seal reduces the leakage flow by one order of magnitude [1,2]. Brush seals also have been found to enhance dynamic performance, cost less and are lighter than labyrinth seals. Even though industrial brush seals have been successfully developed through extensive experimentation [1,2], there is no comprehensive numerical methodology for the design or prediction of their performance [3,4,5]. The existing analytical/numerical approaches are based on bulk flow models [6,7] and do not allow the investigation of the effects of brush morphology (bristle arrangement), or brushes arrangement (number of brushes, spacing between them), on the pressure drops and flow leakage. An increase in the brush seal efficiency is clearly a complex problem that is closely related to the brush geometry and arrangement, and can be solved most likely only by means of a numerically distributed model.

## STATE-OF-THE-ART

---

- ★ The reduced leakage, and physical compliance of the brush body to external perturbing factors are features that stand out in turbomachinery applications where there are expected boundary variations due to mass flow, brush fibers' compliance pressure, temperature, and time dependent eccentric shaft motion. All these characteristics have made the brush configuration an especially interesting and worthy candidate.
- ★ Rolls-Royce(RR), in 1980's, has successfully introduced a brush seal manufactured by Cross Mfg. Ltd.(CML) on a demonstrator engine, and then tested it for several thousand hours, Fergusson[1]. More recently EG&G Sealol, Technetics, Detroit-Allison and others have enabled full programs of study of this type of seal.
- ★ Conclusions of a recent workshop on code development(1992) indicate that while the brush seals works well, there is a need to improve its performance characteristics. Such a goal can be achieved by using cascades of brushes, nonhomogeneous brush morphology, "non-conventional" brush structure design, and in general, a process of optimization of brush design parameters.
- ★ The concept employed by the lumped bulk flow numerical models can not predict local brush compliance, associated local flow phenomena and the pressure drops and the transient effects associated with them. The importance of the local flow phenomena in the sealing process is paramount to the global performance of the brush.

## **CURRENT RESEARCH ACTIVITIES :**

---

- **DEVELOPMENT AND VALIDATION OF A NUMERICAL ALGORITHM AND COMPUTER CODE THAT UTILIZES MATHEMATICAL MODEL WITH DISTRIBUTED PRIMARY PARAMETERS(NAVIER-STOKES EQUATIONS), UNDER NASA GRANT , NASA LEWIS RESEARCH CENTER.**

Computer Code Allows: Estimation of the pressure drops (flow rates) for the typical brush seal segments of different shapes: i.e. bristles diameters, configurations, packing densities.

### **APPROACHES:**

-Large size characteristic segment: 7-10 rows with 10 pins in one row in the transversal direction

-Brush Partitioning: inflow segment(first 3 rows), central part, outflow segment(last rows)

- **DEVELOPMENT OF THE COMPUTER PROGRAM THAT ADDRESSES BRISTLES MOTION AND ITS INFLUENCE ON THE PRESSURE DROPS AND FLOW RATES.**
- **FURTHER MODIFICATION OF THE EXPERIMENTAL FACILITY FOR THE PURPOSES OF CODE VALIDATION. DIFFERENT SHAPES OF THE BRUSH SECTION**

## **OBJECTIVES AND ACCOMPLISHMENTS:**

---

- **Develop verified family of CFD codes for Analyzing Brush Seals**

- idealized(uncompliant) 2D configuration
    - regular gridding
    - variable grid size

✓  
✓  
✓

- compliant 2D geometry

under development

- **Experimental Facilities for the Adequate Code Verification**

- stationary bristles(cylinders)

✓

- moving bristles

under development

- **Qualitative and quantitative analyses of the Fluid Flow in the Brush Seal Configuration**

- flow around one bristle, level 1
  - flow around several bristles, level 2
  - flow in deep tube bundles, level 3  
(intermediate pitch-to-diameter ratio)  
7 rows of pins with 11 cyl. in a row

✓  
✓  
✓

-----

- flow through uncompliant brush prototypes  
(small pitch-to-diameter ratio), level 4

✓

- flow through the characteristic brush  
segments, brush partitioning technique  
level 5

✓

## PROBLEM DEFINITION

---

Conclusions of a recent NASA Seal Workshop[5] indicate that while the brush seals work well, there is a need to further improve the performance characteristics. Such a goal can be achieved by using cascades of brushes, nonhomogeneous brush morphology, "non-conventional" brush structure design, and in general, a process of optimization of brush design parameters[18]. The distributed velocity fields( $u,v$ ) and the associated pressure maps are of vital importance for the prediction of the average pressure drop, or the possible sudden failure of the brush seal under unexpected local "pressure hikes". The momentum carried by these velocities(or the upstream pressure) can force the brush deformation, and can create favorable conditions for the brush 'opening', followed by seal failure[4,15]. It is in this context that the development and validation of a numerical model with distributed primary parameters( $u,v,p$ ) becomes important.

The design goals of the model are to determine the pressure drop for a configuration specified by the designer, i.e. the density of the brush packaging, length, number of rows, bristles sizes(homogeneous or nonhomogeneous), distances between the rows, or brushes respectively(single, double or cascade brush). The systemic goals are a) to develop physically relevant packages of assumptions for the simulation of the brush seal and to implement a robust numerical method(using primitive variables  $u,v,p$ ) for the calculation of the forced convective flow through dense brush-like cylinder arrays, and b) to analyze numerically different aspects of the flow dynamics in the generic brush prototypes. Achieving the goals set forth in a) and b), will allow usage of predictive design codes with a high level of reliability.

For analysis and classification purposes we identified four different models of the generic brush geometry[19]. The simplest

model(Level 1) assumes flow analysis in the vicinity of a non-moving single brush bristle[20]. The second level(Level 2) model introduces the analysis of a limited cluster that consists of several non-moving bristles[19,20], that do influence jointly the flow field through disturbances generated by the wake vortices. The next level(Level 3) introduces the analysis for a multi-cluster[21], and finally we assemble large numbers of clusters with small pitch to diameter ratio(PTDR) that actually simulate the real brush(Level 4, [22] ). Each one of these levels is designed to introduce one additional level of difficulty, help learn more about the physics of the flow, and increase the level of confidence in the final numerical model.

One can see an idealized schematic of a linear brush seal in Fig. 1. In real conditions the flow upstream of the seal exhibits both circumferential and axial components. Reynolds numbers that are typical for the circumferential component are usually in the range  $10^4$ -  $10^6$ , while the Reynolds number of the transversal component (leakage flow) does not exceed low laminar values that are defined by the design requirements of the seal(an ideal case leakage is equal to zero). A review of the data published by Chupp et al[6], Nelson and Chupp [10], Dowler[11] allowed us to determine the typical ranges of the parameters that are usual for the brush seals tested by the industry. The level of the leakage flow and maximum velocity in the pitch between the bristles were estimated as 3.17 m/s and 76.55m/s respectively. In Fig. 2 one can find an approximate range of Reynolds numbers typical for a brush seal functioning in air. As we can see from this qualitative figure an assumption of laminar and incompressible fluid can be well justified since the Reynolds numbers are not in the turbulent range and the Mach number is  $M < 0.4$ . The authors have established through their experimental work[4,15] that the major factor contributing to the pressure drop is the longitudinal flow in the X direction in the XOY plane(Fig. 1). Thus, in order to simplify the model we assumed a two dimensional flow, and neglected both curvature effects and flow in the Z direction.

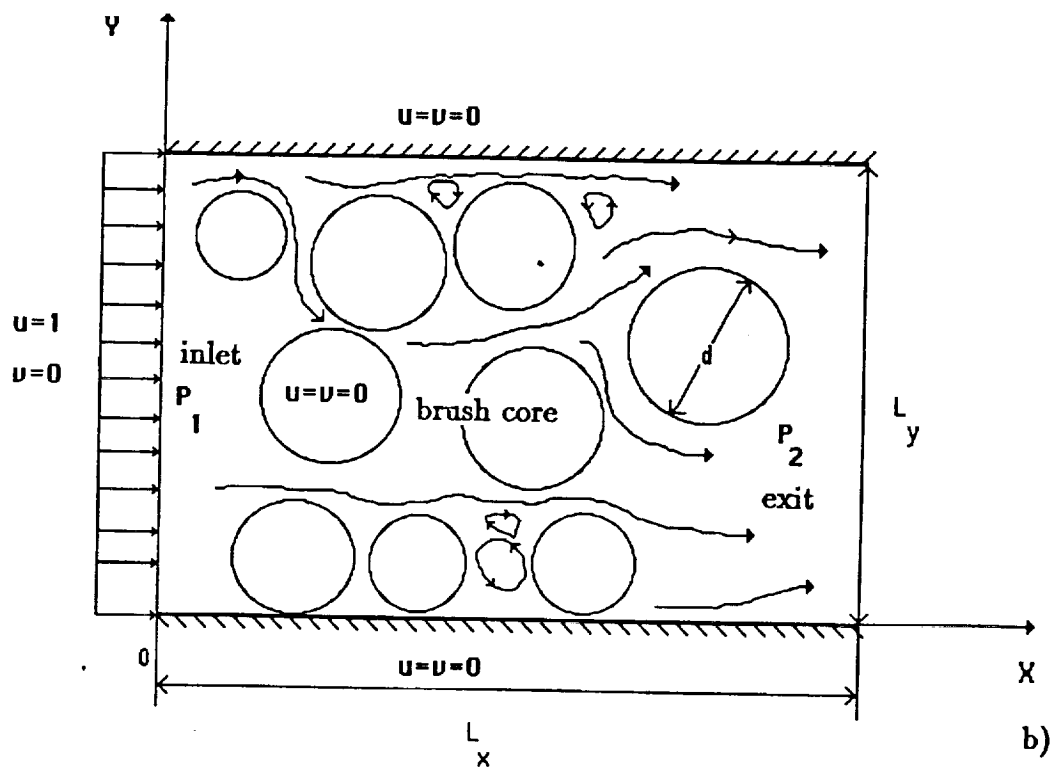
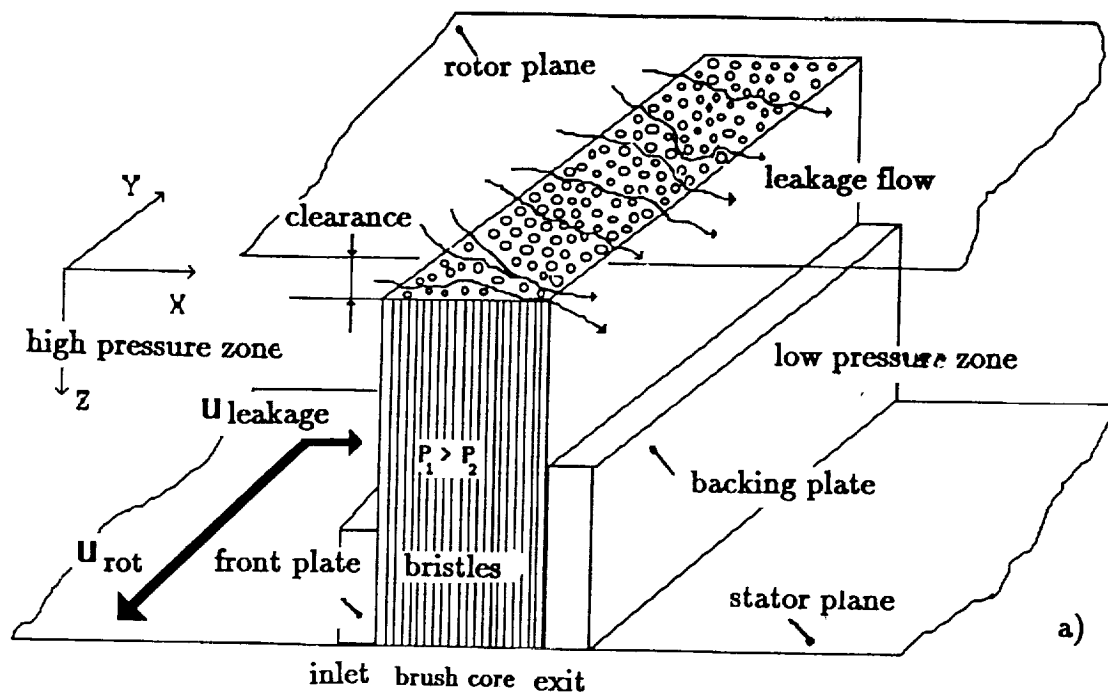


Figure 1(a,b) Idealized Schematic of the Brush Seal and Flowpath

a) generic seal

b) problem definition

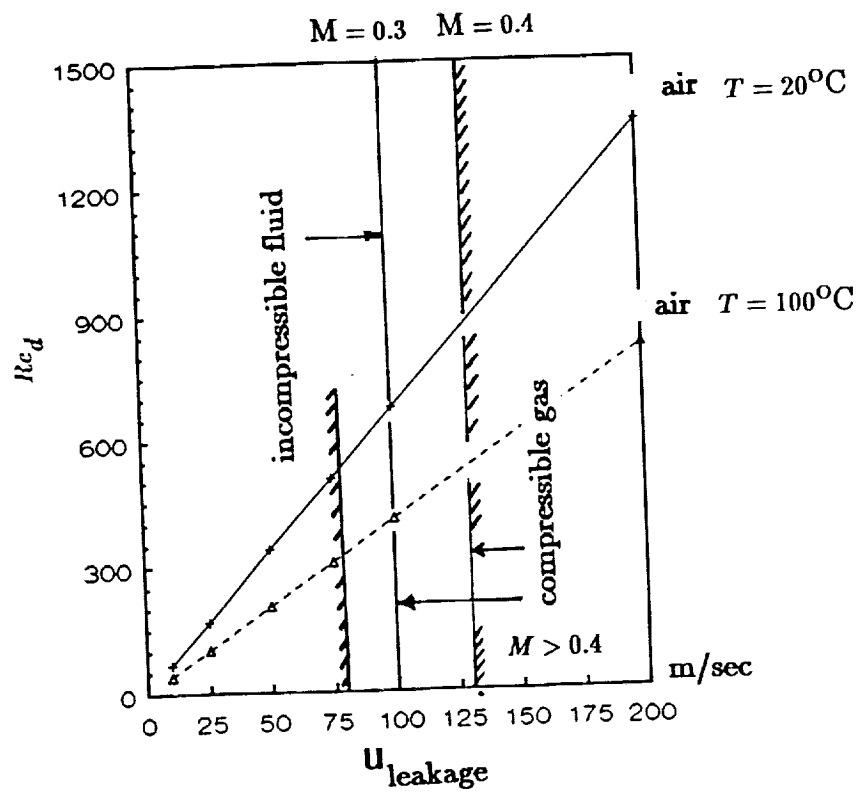


Figure 2. Qualitative Diagram of the Flow Leakage vs Reynolds Number



## MATHEMATICAL MODEL

---

### • Characteristic Geometry

The computational domain (with  $L_x$  as length and  $L_y$  as width) is represented by the horizontal brush cross section restricted by two walls as shown in Fig. 1b. Inside the domain, solid bodies (bristles) of round cross-section are located, thus, creating a structure of (n) rows with (m) elements in each row. The bristle diameter is used as a characteristic length scale ( $L_0$ ) and the velocity at the entrance, as a characteristic velocity ( $U_0$ ).

### • Governing equations.

It is assumed that the flow is viscous and laminar and it is caused by the pressure difference across the generic brush. The initial velocity distribution  $u(x,y)$  and the characteristic pressure at the entrance are assumed known. The influence of the body forces is excluded from the problem definition. We also neglect the influence of heat transfer on the flow structure (isothermal flow). The two-dimensional Navier-Stokes equations for unsteady incompressible viscous flow, can be written in dimensionless conservative form (Cartesian), as

$$\frac{\partial u}{\partial t} + \frac{\partial(uu)}{\partial x} + \frac{\partial(uv)}{\partial y} = -\frac{\partial P}{\partial X} + \frac{1}{Re} \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \quad (1)$$

$$\frac{\partial v}{\partial t} + \frac{\partial(vv)}{\partial y} + \frac{\partial(uv)}{\partial x} = -\frac{\partial P}{\partial Y} + \frac{1}{Re} \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) \quad (2)$$

$$\begin{aligned} \nabla^2 P = & -\frac{\partial^2(u^2)}{\partial x^2} - 2\frac{\partial^2(uv)}{\partial x \partial y} - \frac{\partial^2(v^2)}{\partial y^2} + \left\{ \frac{\partial \vartheta}{\partial t} + \right. \\ & \left. + \frac{1}{Re} \left( \frac{\partial^2 \vartheta}{\partial x^2} + \frac{\partial^2 \vartheta}{\partial y^2} \right) \right\} \end{aligned} \quad (3)$$

$$\vartheta = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \quad (4)$$

## MATHEMATICAL MODEL

---

During calculations,  $\vartheta$  (the dilation term) represents the residual that has to shrink to zero if continuity is to be satisfied[23,24]. Previous numerical work of these authors[19,20,21] explored models of Levels 1, 2, and 3 in order to obtain qualitative information about fluid flow in the generic brush configurations.

The new technique that involves brush partitioning proposes the division of the brush in three typical areas(Fig.1): (i)inflow area with the free flow and first several rows of pins with developing flow profile, (ii) a central core that consists of several rows of bristles with symmetric fully-developed flow distribution, and (iii) an outflow zone that includes the last rows of bristles and the area behind the brush where vortices generation is taking place.

Hendricks[25] showed that the first three and last three rows of bristles are usually deflected outward of the main brush core during the experiments. These bristles are located at a considerable distance from each other and thus do not significantly contribute to the overall pressure drop. Since the central core has the highest level of packing density and flow resistance, as a first step we will concentrate our analysis on this region. In general, the total pressure drop in cylinder arrays can be expressed as following:

$$\Delta P = C_z \sum \Delta P_{av} \quad , \quad i=1., \dots, N_{rows}^{(5)}$$

According to Zukauskas et al.[18],  $C_z=1$ , if the number of rows is greater than three. One can conclude that in terms of the pressure field the flow is fully developed as it passed the first three rows.

Assumption of a fully developed flow distribution in the central brush core reduces the size of the computational domain to a characteristic cell of several sequential rows(characterized by a constant  $\Delta P_{av}$ ), if indeed, one supplies proper symmetric conservative boundary conditions.

## BOUNDARY CONDITIONS

---

### • Brush Entrance Region

We assume an entrance velocity profile with  $u=f(y,t)$  and  $v=f(y,t)$ , which in most cases can be reduced to  $u(x=0)=1$  and  $v(x=0)=0$ . In the case of lateral solid walls boundary conditions are specified as non-porous and non-slip, i.e.  $u=v=0$ . In the case of flow symmetry, boundary conditions at the "top" and "bottom" boundaries are given as

$$a) \quad \frac{\partial u}{\partial y} = 0 \quad \text{and} \quad \frac{\partial v}{\partial y} = 0 \quad (6)$$

$$b) \quad \frac{\partial u}{\partial y} = 0 \quad \text{and} \quad v = 0 \quad (7)$$

Exit boundary conditions can be specified in a similar way:

$$\frac{\partial u}{\partial x} = 0 \quad \text{and} \quad v = 0 \quad (8)$$

### • Brush Core Zone

Inflow velocity condition for this zone are the outflow conditions for the brush entrance zone. In this case:

$$v = 0 \quad \text{and} \quad u = f_1(y,t), \quad (9)$$

where  $f_1(y,t)$  is established from the solution of the Navier-Stokes equations in the entrance region. At the outflow of the computational domain we apply Eq. 8, and at the lateral walls Eqs. 6-7.

## BOUNDARY CONDITIONS

---

### • Brush Outflow Zone

Inflow velocity conditions for this area are calculated via the solution of the Navier-Stokes equations at the brush core zone. That implies  $v=0$  and  $u=f_2(y,t)$ . At the lateral walls we employ conditions (6-7). The exit flow is allowed to develop naturally based on boundary conditions situated far downstream. The velocity boundary conditions imposed at the exit are derived from (i) the satisfaction of the continuity equation (specifically in the direction of the U velocity),

$$\frac{\partial U}{\partial x} = - \frac{\partial v}{\partial y} \quad (10a)$$

and (ii) the condition for fully developed velocity gradient for the vertical component

$$\frac{\partial v}{\partial x} = 0 \quad (10b)$$

### • Boundary Conditions for the Poisson Equation

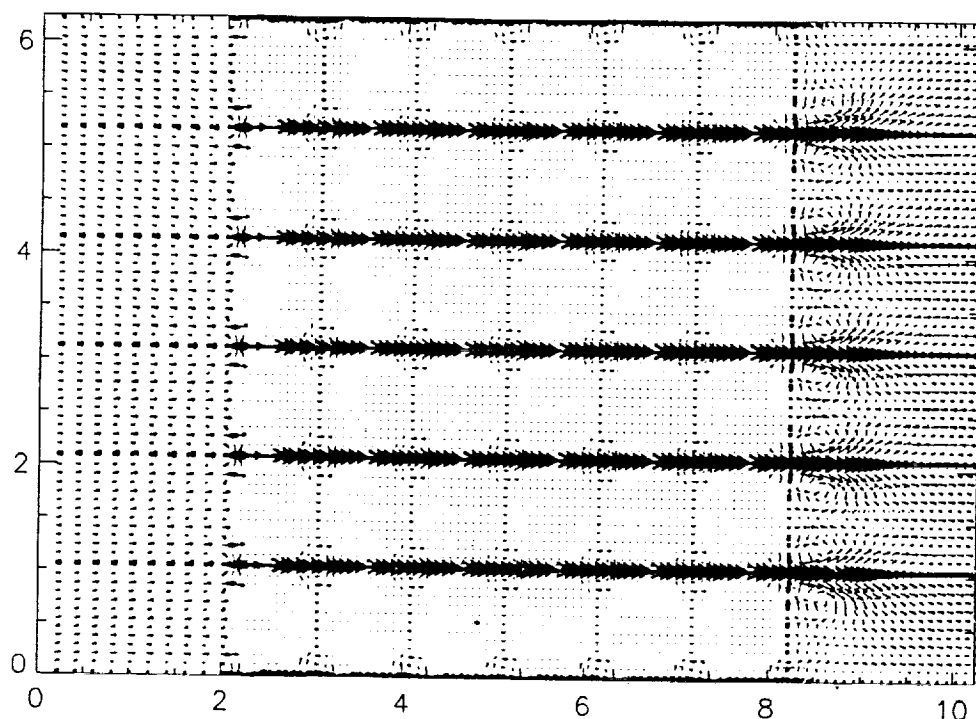
The dynamic pressure P, is determined from the balance of the normal forces with the inertia and viscous forces. This formulation implies Neumann type boundary conditions. The effects of the terms  $\frac{\partial u}{\partial t}$  and  $\frac{\partial v}{\partial t}$  have been considered negligible. The following formulation of Eqs. 11 and 12 is totally independent of the boundary configuration, and thus applicable with no restrictions.

$$\frac{\partial P}{\partial x} = - \left[ \frac{\partial(uu)}{\partial x} + \frac{\partial(uv)}{\partial y} \right] + \frac{1}{Re} \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right); \quad (11)$$

$$\frac{\partial P}{\partial y} = - \left[ \frac{\partial(uv)}{\partial x} + \frac{\partial(vv)}{\partial y} \right] + \frac{1}{Re} \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) \quad (12)$$

In combination with the Dirichlet conditions for the velocity, Eqs. 11 and 12 form a well-posed boundary-value problem with resulting solutions free of odd-even velocity-pressure decoupling as described by Gresho[26] and Patankar[27].

### Odd-even Velocity-Pressure Decoupling For the Inline Brush Segment



Initial conditions( $t=0$ ). The input velocities are given as  $u=1$ ,  $v=0$ . The pressures are set initially to an arbitrary, operator chosen  $P=P_{\text{ref}}=\text{const.}$

## SOLUTION PROCEDURE

---

The discretization of the system of governing equations introduced above, follows the use of the Alternating Direction Method[28] applied to a collocated grid. The procedure uses the full direct approximation of each term within the differential equation on every half time step,  $\Delta\tau/2$ . One obtains the following system of linear algebraic equations.

The spatial derivatives, with the exception of the convection terms and cross-derivatives, are approximated by an implicit second order central finite difference. For the convective terms, the implicit form of the third-order deferred correction scheme proposed by Kudriavtsev[30] and Hayase et al.[31] was implemented[4].

After discretization of the boundary conditions we obtain a self sufficient system of linear algebraic equations that is solved using a tridiagonal matrix elimination. The steps of the solution are as follows:

- introduce the initial flow and pressure field at the  $n$  time level
- solve in the  $x$  direction for  $U$  velocity at the  $n+1/2$  time step
- solve in the  $y$  direction for the  $U$  velocity at the  $n+1$  time step
- solve in the  $x$  direction for  $V$  velocity at the  $n+1/2$  time step
- solve in the  $y$  direction for the  $V$  velocity at the  $n+1$  time step
- solve the Poisson's pressure equation at the  $n+1$  time step by means of the pseudo-transient method within the set of internal ADI iterations  $n_{it}=1, \dots, n_f$ : (i) in the  $x$  direction at the  $s+1/2$  pseudo-time step and (ii) in the  $y$  direction at the  $s+1$  pseudo-time step
- advance to the next time level .

On each iteration pressure at the reference point  $P(1,2)$  is assigned as  $P^*$ . Nondimensional pressure at every node is calculated as:  $\Delta p_{i,j} = P^* - P_{i,j}$ . If steady-state solution is of interest we employ pseudo-transient method as proposed by [28].

## RESULTS AND DISCUSSION

---

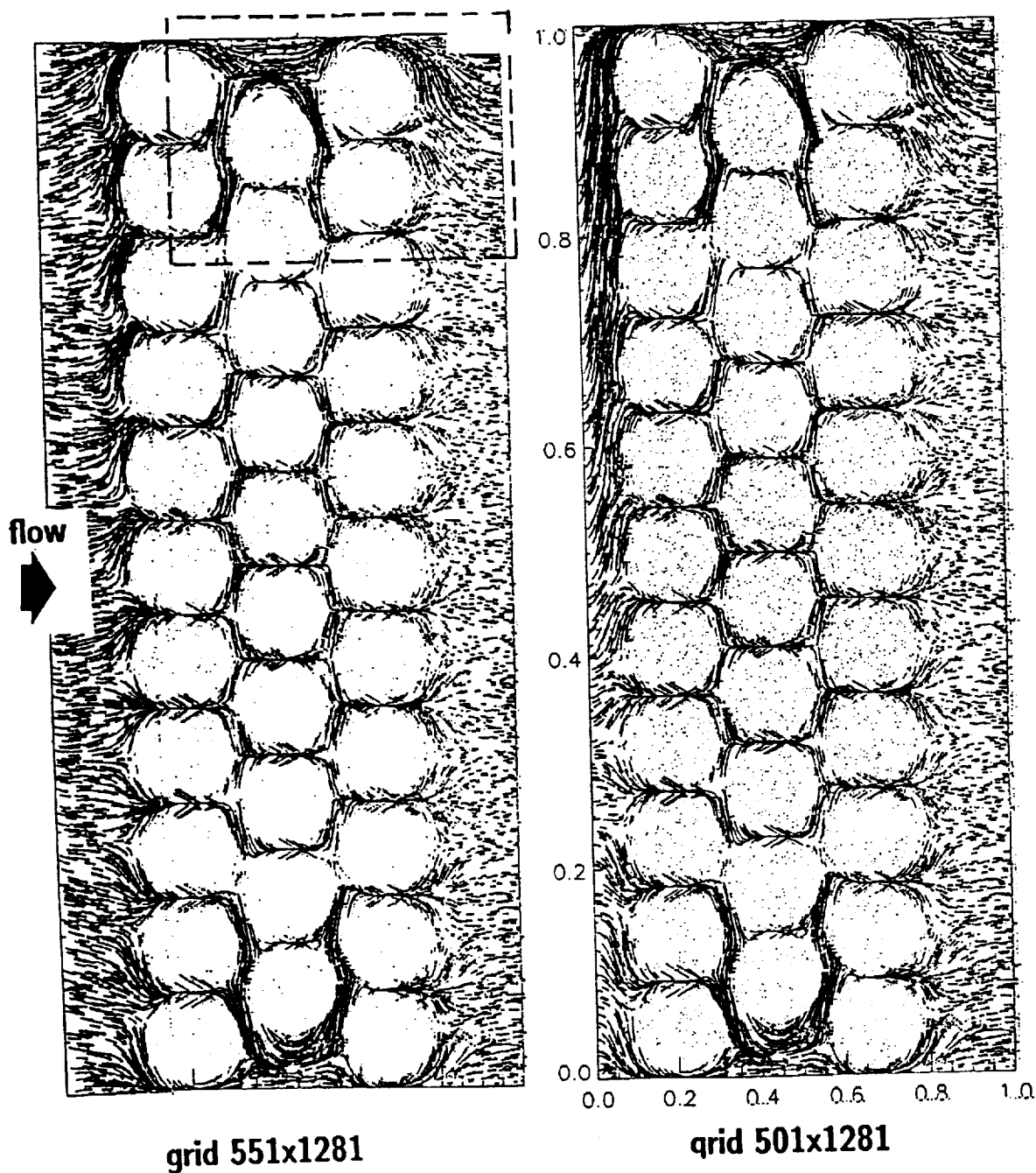
- **Pressure and flow patterns for different generic brush formations.**

There is a considerable amount of numerical and experimental work treating the flow around circular or squared cylinders in crossflow. However, only few investigations concentrated their attention on the flow interaction with the tube bundles[32,33,34]. The effects of the cylinders' arrangement and the array size, or morphology on the flow structure and pressure drops has not been extensively studied, especially if the pitch-to-diameter ratio(PTDR) is smaller than 1.

- Braun et al.[21], Kudriavtsev et al.[20], and Kudriavtsev[19] have studied systematically the time-flow, and time-pressure development in arrays of cylinders with PTDR equal and smaller than 1. The database that emerged, showed that the development of flow and pressures around one cylinder, small groups of cylinders, and large groups of cylinders, do not lend themselves to extrapolation from one configuration to another. This is exemplified in Figs. 3 and 4. Figure 3 presents the pressure oscillations at the trailing edge of a single cylinder located in the square channel of Fig. 1. On inspection, it can be seen that in a quasi steady regime the pressures in the wake of the cylinder reach a repetitive oscillatory profile within an envelope of  $\pm 5\%$ . When the number of cylinders is increased to five, arranged in a staggered formation of two rows, the same envelope is contained between  $\pm 10\%$ , Fig. 4a. Finally, the increase of the cluster to 72 cylinders, arranged also in a staggered formation of seven rows, generates in the wake of the array oscillatory pressures that can be contained within  $\pm 2.5\%$  envelope, fig. 4b. These results, when considered in conjunction with the flow patterns already discussed by the authors in previous papers, demonstrate that phenomena indigenous to small arrays do not, as a rule, apply to large formations. Therefore one has to be extremely careful during the analytical/numerical modeling of large arrays. Special methodologies, that involve realistic treatment of

# PARAMETRIC STUDIES OF THE FLOW IN THE BRUSH PROTOTYPES LOCATED IN THE OIL TUNNEL

$Re=195$



## CALCULATED FLOWFIELD

$\Delta P(\text{num}) = 4.31 \text{ psi}$     $\Delta P(\text{exp}) = 4.02 \text{ psi}$  , error = 7%



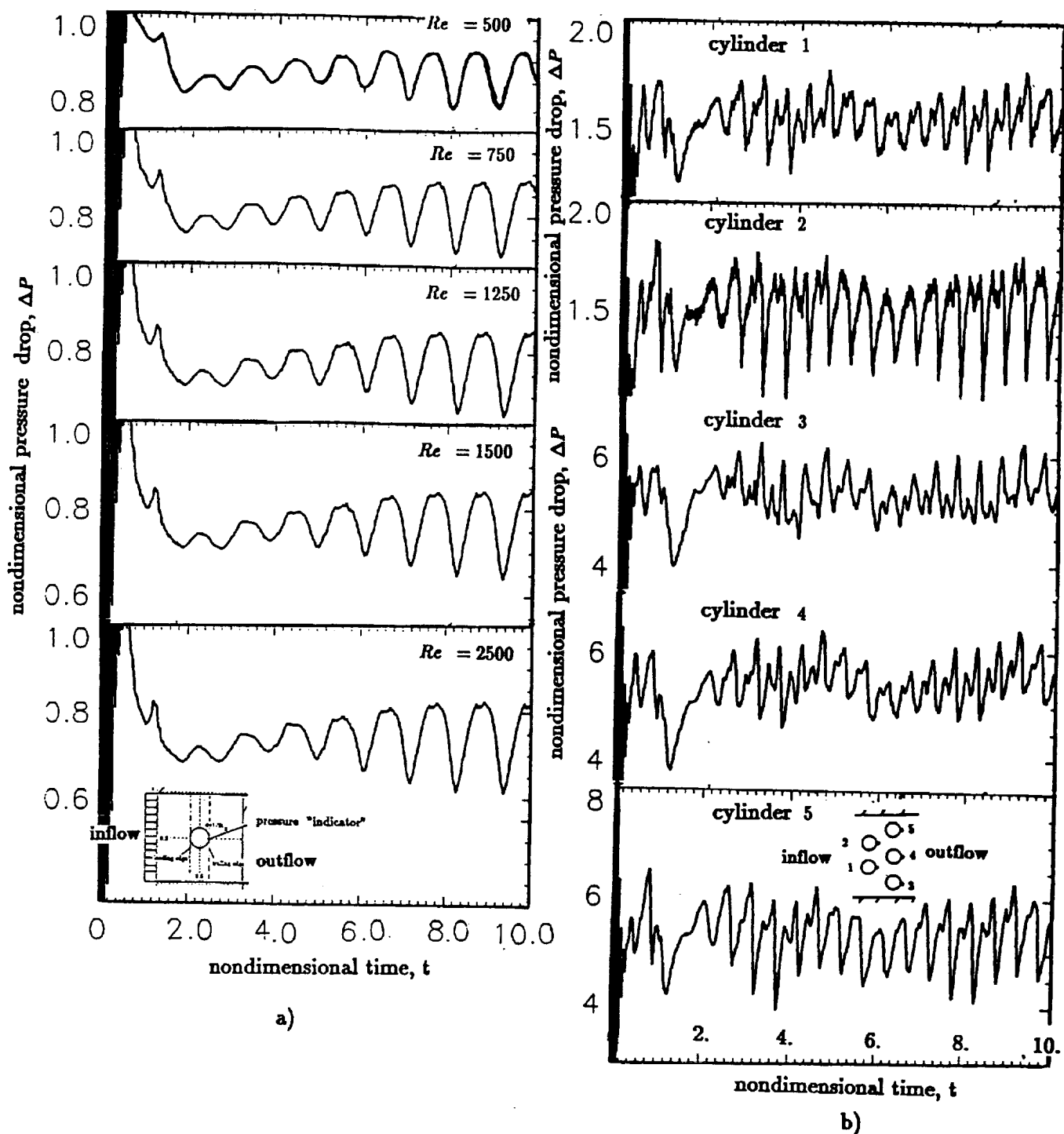


Figure 3(a,b) Pressure Development Behind Single Cylinder(Level 1)  
and Small Cluster of Cylinders(Level 2)

a)single cylinder in the square channel

b)small cluster in the square channel

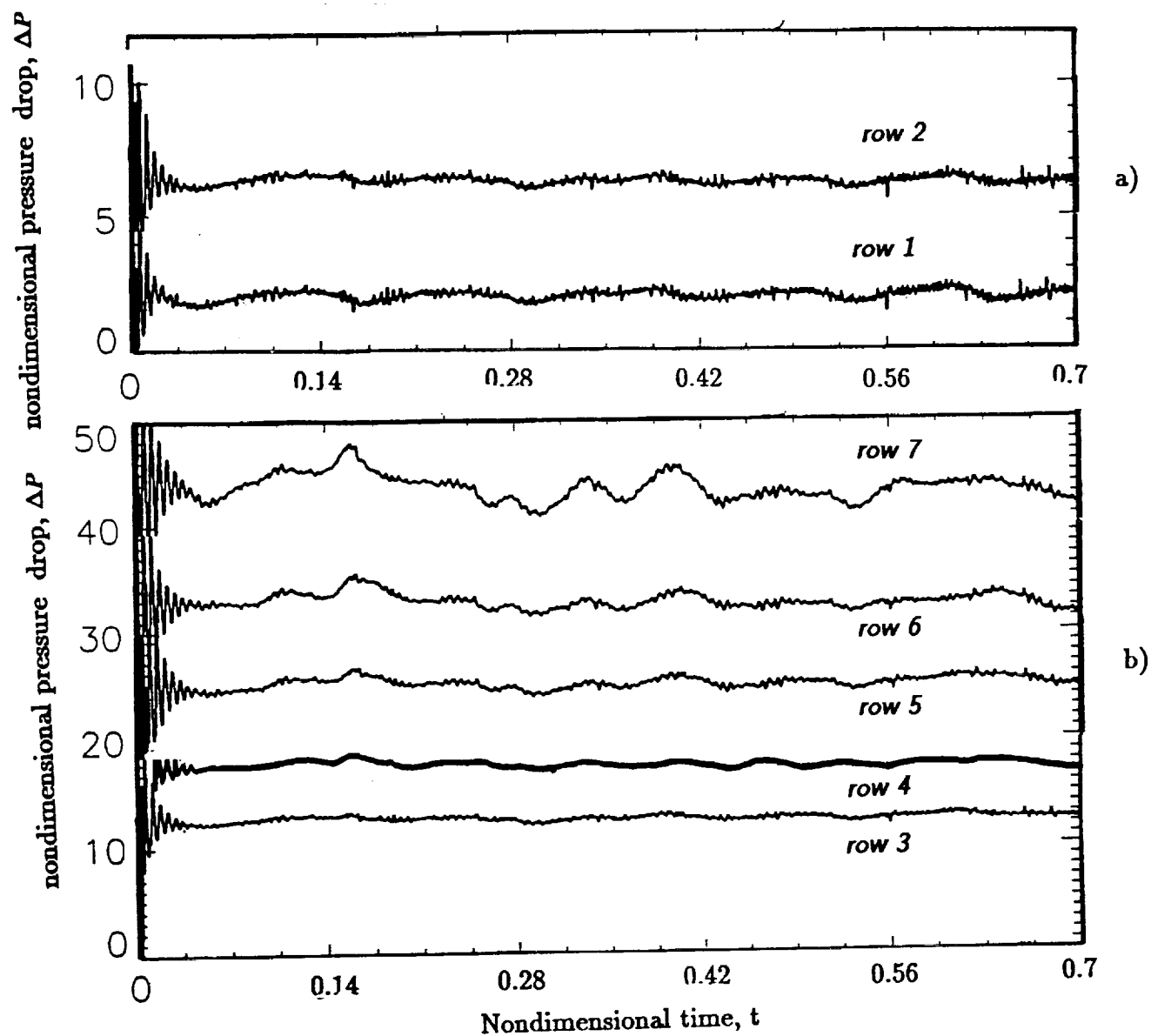


Figure 4(a,b) Pressure Development Behind a Large Cluster of Cylinders(Level 3)

(Locations of the "pressure indicators" are shown in Fig. 5)

boundary conditions, have to be applied to small arrays, that can then be assembled (based on these boundary conditions) in large clusters, and render a realistic reproduction of the flow conditions.

Figures 5 present the flow development in an array of 7(rows)x11 round pins with  $PTDR_L = PTDR_T = 1$ . Along the walls, one can clearly observe high velocity rivering jets that engender contiguous regions of lower pressure which attract the flow from the center of the generic brush[21]. The central region appears to be, fortunately, quite repetitive in nature, and indicates that the use of a repetitive cell for the construction of a large brush core is a feasible alternative. Finally the wake region presents the formation of the exit jets, and the incipient formation of meandering vortices.

The same algorithm used to obtain the results presented in Fig. 5( $PTDR=1$ ) has been used for the modeling of the dense array of Fig. 6. The flow, at  $Re=195$ , presents a steady-state pattern in this array of three rows with  $PTDR_L = PTDR_T = 0.084$ . This packing density is of significance for the real industrial brush seal applications( $1/10 \geq PTDR \geq 1/30$ ). Modeling of the fluid flow through the pin array with such a  $PTDR$  is a challenging computational task that so far has not been addressed in the open computational fluid dynamics(CFD) literature. The flow is dominated by accelerating-decelerating streams formed by the system of converging-diverging nozzles(constituted by the adjacent cylinders). These flows are characterized by extremely high convective gradients at their smallest cross-section. This situation imposes limitations on the computational algorithm in terms of computational stability, resolution and global accuracy. The window marked in Fig. 6a, is shown in detail in Fig. 6b. One can get an appreciation of the complicated flow formations that are at work. In Fig. 6b the rivering, lateral rivering, and the nozzle-jets are quite evident. The formation of the jets at the exit of the array can also be observed. The corresponding experimental results and the successful superposition of these results and the numerical simulation are shown in this sequence(Fig. 6b). For an entire array

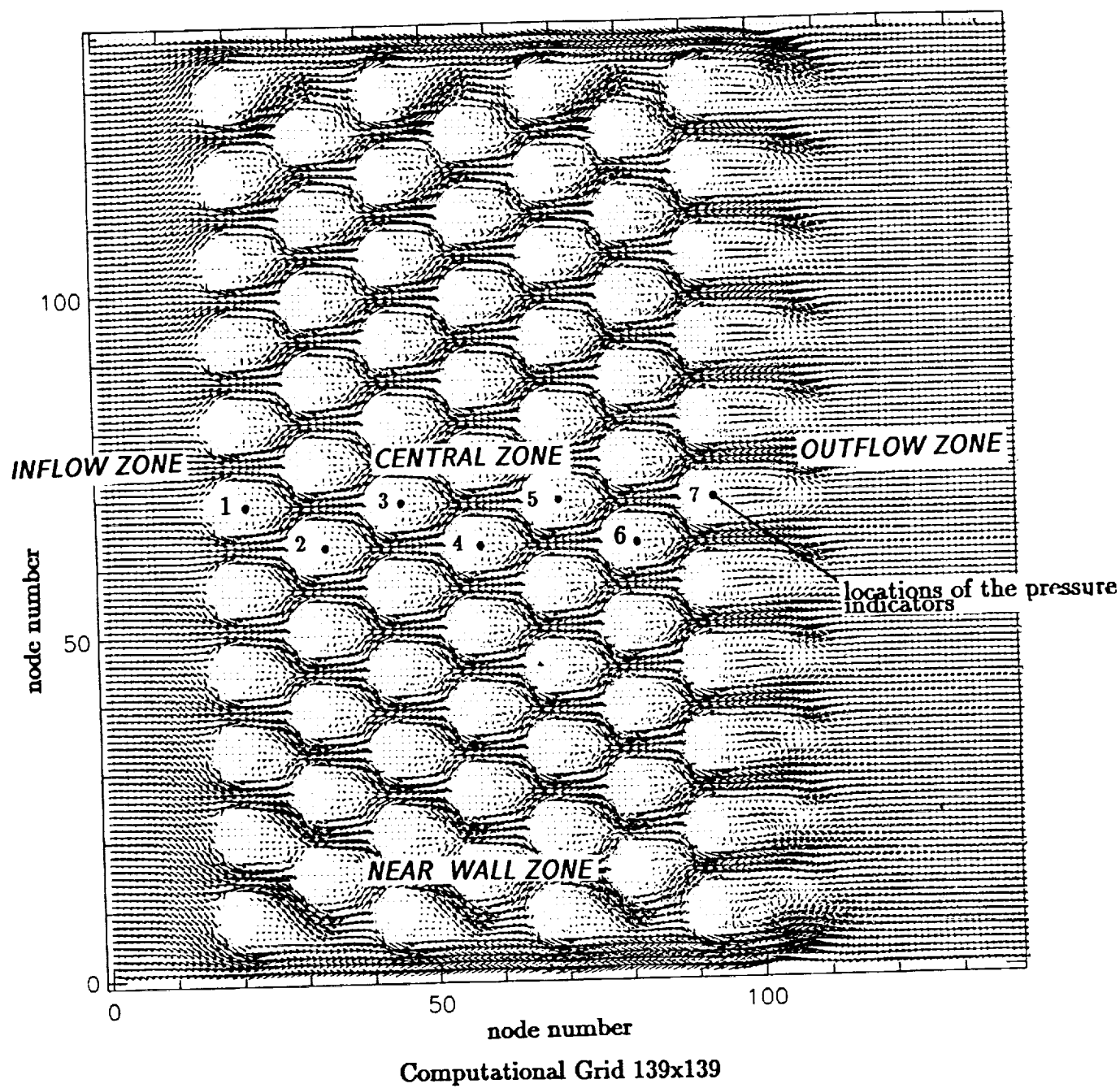


Figure 5. Typical Flow Structure in the Large Cluster of Cylinders(Level 3)

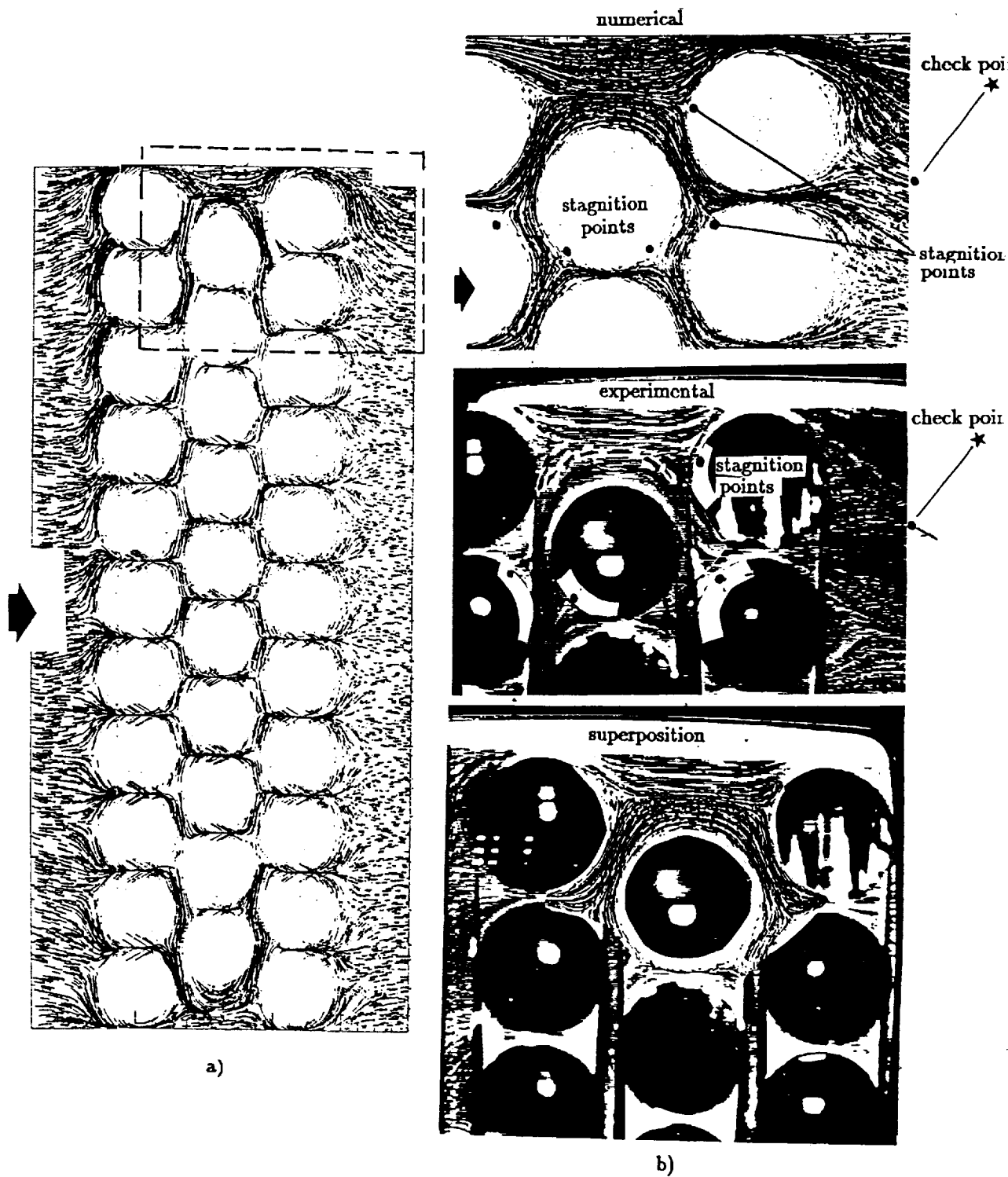


Figure 6(a,b) Large Cluster of Densely Packed Cylinders at  $Re=195$ (Level 4)

a) flow through the cylinder's array

b) detail of flow near the wall marked at Fig.6a .

Comparison of numerical and experimental results.

of 3(rows)x11, Fig. 7a, presents experimentally the interaction of 12 convective jets as they interact with each other and with the wall. The experiment was run at  $Re=1000$ , and a rather unusual flow formation was observed in the wake of the array. The low pressure region that developed near the walls, in- and outside of the pin array, causes constant jet flow deviation towards the walls. The void created in the wake is associated with a pressure that is lower than the far downstream pressure. This pressure field engenders a recirculation bubble that occupies considerable part of the outflow cross section and consists of the two butterfly like vortices with strong "negative" flow along the centerline of the cross section. The global resulting recirculation structures include numerous vortices and require high computational accuracy in order to obtain any realistic solution. Such solution can be observed in Figs. 7b and 7c. For the numerical results of both Figs. 6 and 7 it has been found that the flow structure, inside the array, converged to a steady solution much faster than the flow in the wake of the array, Figs. 7b, 7c.

For  $PTDR < 0.1$ , the numerical solution can be obtained only by large scale CFD modeling that requires grid sizes approaching  $1000 \times 1000$  nodes. Thus a further increase in the size of the array or any further grid refinement becomes rather prohibitive, and appropriate strategies need to be found to handle this situation.

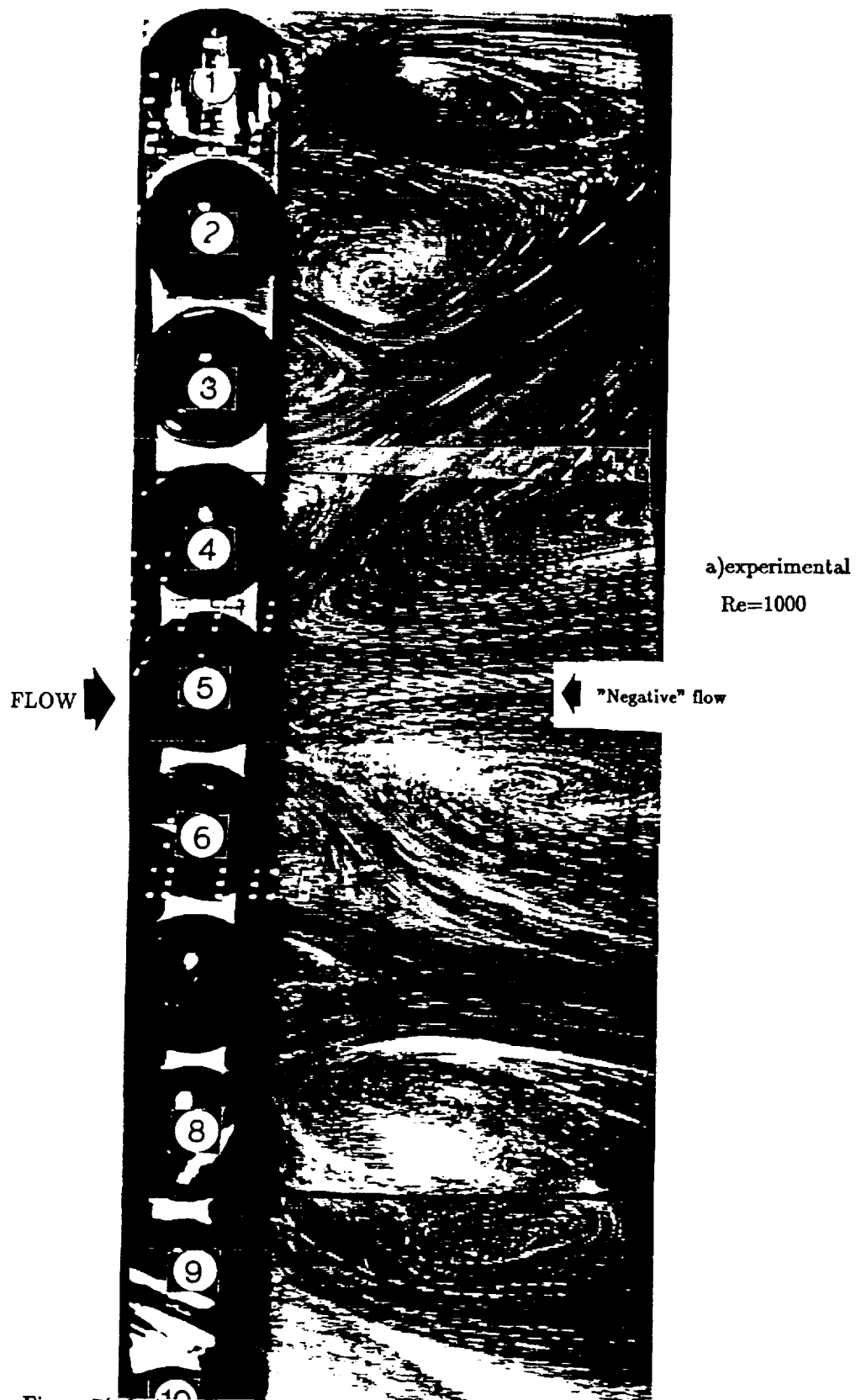
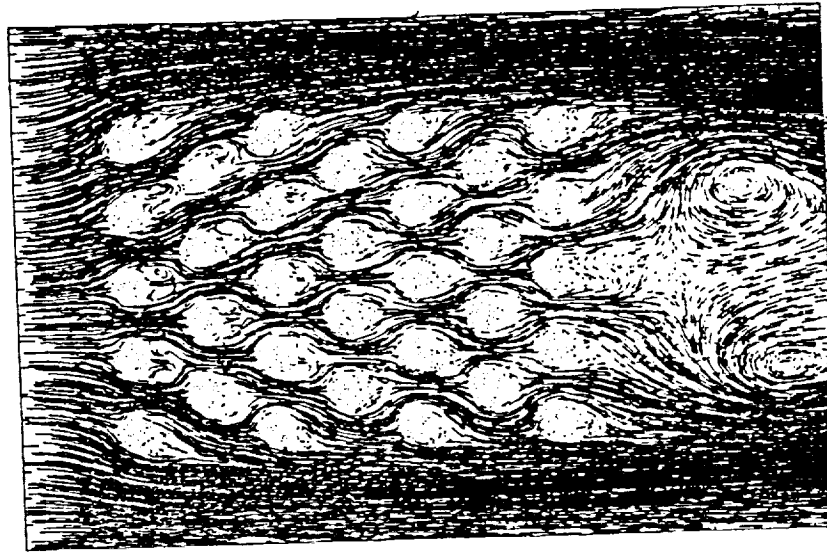
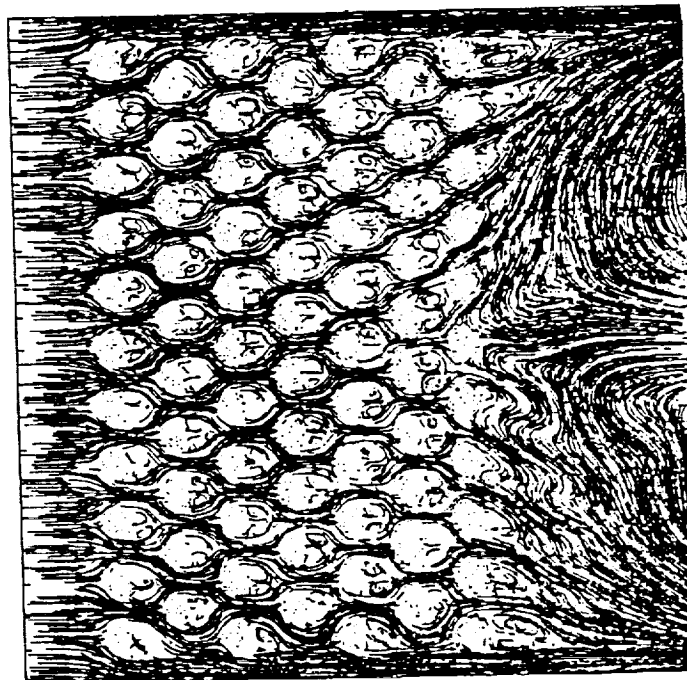


Figure 7(a,b,c) Butterfly Formations in the Wake of the Cylinder's Clusters



b)numerical(small array)



c)numerical(large array)

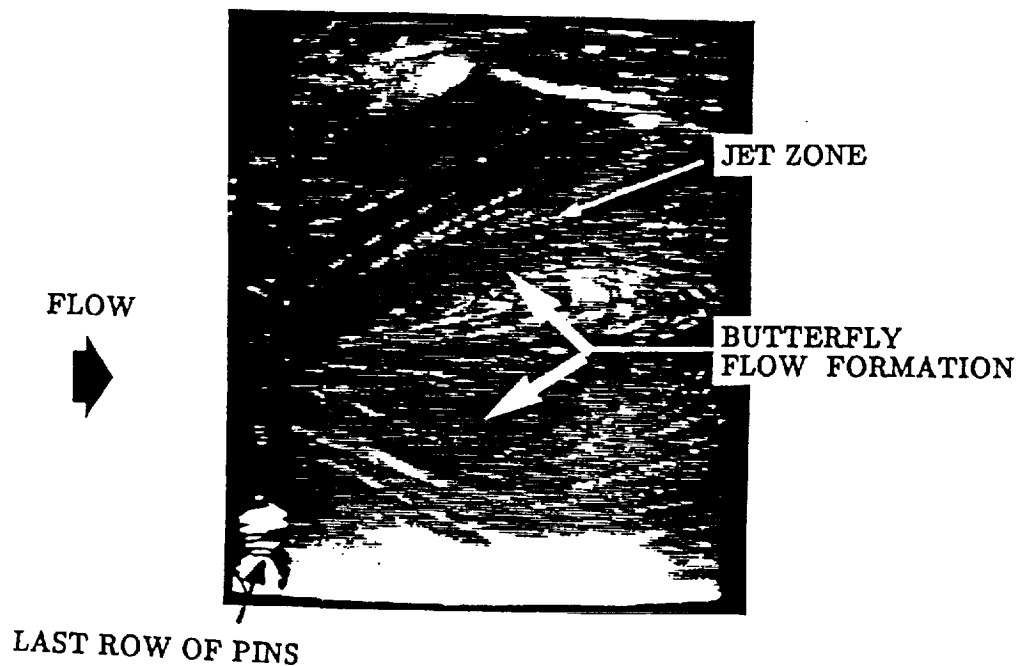
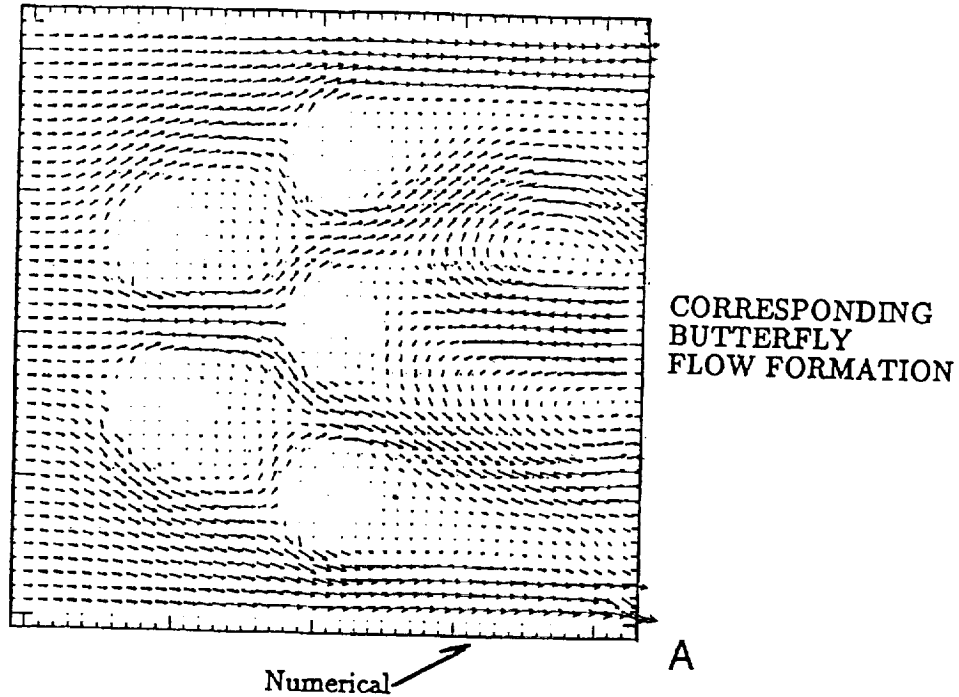
Figure 7(a,b,c) Butterfly Formations in the Wake of the Cylinder's Clusters

$Re=1000$



## PARAMETRIC STUDIES OF THE FLOW IN AN ARRAY OF FIVE STAGGERED CYLINDERS

- authors have extended their analysis to a representative cell of five staggered cylinders. The longitudinal and transversal pitches are equal to each other,  $PTDR_L = PTDR_T = 1$ , while the cylinder diameter is  $d = 0.143L_0$ .



## **The new approach of brush segmentation and assembly**

---

The totality of the numerical and experimental work presented above has led us to the conclusion[19], that we were fastly approaching the limits of our modeling capability from the standpoint of the computer capacity to handle the large arrays necessary to model a full brush. We have learned that

- a) the flow inside the brush reaches a convergent solution, much sooner than near the wall boundaries or in the brush wake;
- b) in a deep cluster the flow is practically repetitive, as long as it is at least two rows away from the inlet or exit of the generic brush;
- c) the pressures fall monotonously in the core portion of the simulated brush[4];
- d) the computational time for calculating flow in dense arrays of cylinders(3rows x 11 lines) at moderate Re numbers can become prohibitive.

As mentioned earlier, the brush can be partitioned into several areas. The numerical simulations shown in Figs. 8 explain the rationale of the brush partitioning approach. One of our previous conclusions was that large brush modeling is restrictive, so one tries to reduce the computational domain to a representative cell. In order to define optimally this cell, and its environment, two geometrical setups were chosen. First is the flow in the channel with non-slip walls as boundary conditions. The second setup replaces the wall with symmetry conditions at the boundary. From Fig.8(a,b) one can see that the wall introduces near wall effects that considerably change the flow structure. Thus one has to select the symmetry boundary conditions setup, as the optimal one(see Eqs.7-8,10). At  $Re=100$  in the later time stages of the flow development one observes the incipience of an asymmetric mode in

the outflow zone. This situation induces asymmetry in the central part of this brush segment(Fig. 8b and 8d). In larger(deeper) cylinders bundles this effect does not penetrate to the core(like in Fig. 5). Thus, the core and the outflow segments need to be treated as separate computational domains.

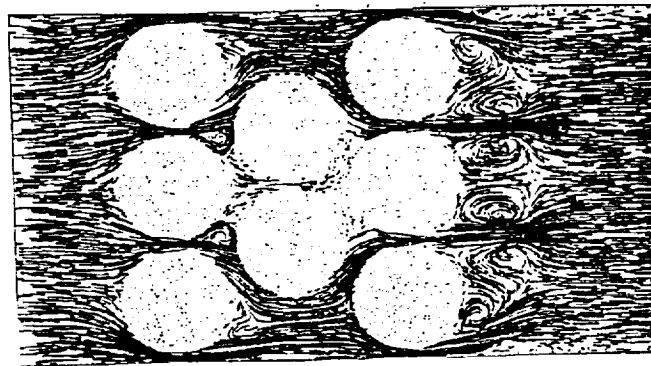
To respond to these physical conditions and the conclusions drawn above in items a) to d) we proceeded to assemble a full brush out of the component segments that were identified in the mathematical model section. The continuity and momentum governing equations remain the same, but the boundary conditions had to be modified in order to take into the account the repetitive nature of the assembly of the brush core. These boundary conditions were described by Eqs. 6-10. For the verification of the concept we have used a partitioned array that contains the segments shown in Fig. 9. Figure 9a, presents a succession of identical core segments that are characterized by the same pressure drops. Figure 9b displays the inflow and outflow segments that are added to the central core in order to form a complete system. The resulting total pressure drop can be calculated as

$$\Delta P_{\text{brush}} = \Delta P_{\text{inflow}} + N^{\text{seg}} \Delta P_{\text{central}} + \Delta P_{\text{outflow}} \quad (19a)$$

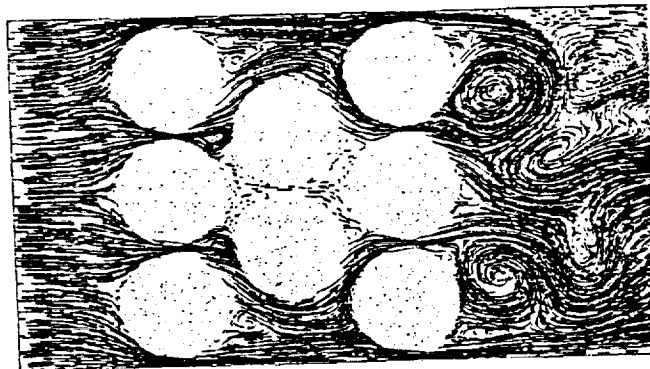
If one considers the Hendricks[25] report, and if the brush has more than  $n \geq 10$  rows, it seems rational that the overall pressure drop can be evaluated with little error by considering only what we defined as the core portion. Thus one can simplify Eq. 19a to

$$\Delta P_{\text{brush}} = n \Delta P_{\text{central}}^* \quad (19b)$$

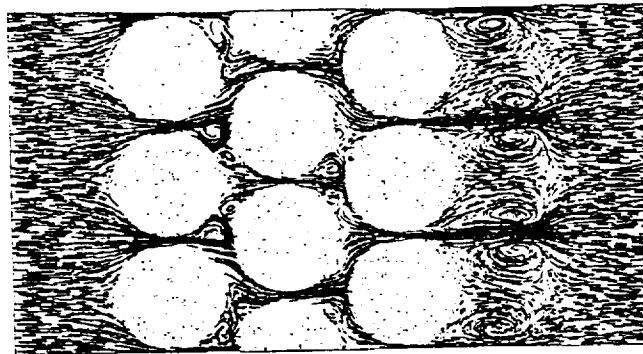
where  $\Delta P_{\text{central}}^*$  - average pressure drop per single row.



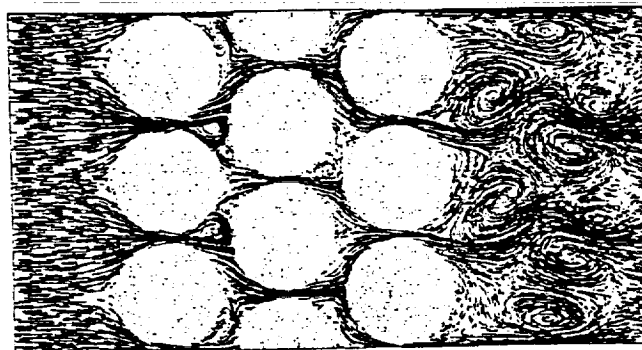
a)



b)



c)



d)

Brush Segment Located  
in the Channel

Symmetry boundary condition  
on lateral boundaries

Figure 8(a,b,c,d) Samples of the Fluid Flow in the Brush Segments at  $Re=100$   
 a)symmetric mode(initial stage of flow development)  
 b)asymmetric mode(final stage of flow development)  
 c)symmetric mode(initial stage of flow development)  
 d) asymmetric mode(final stage of flow development)

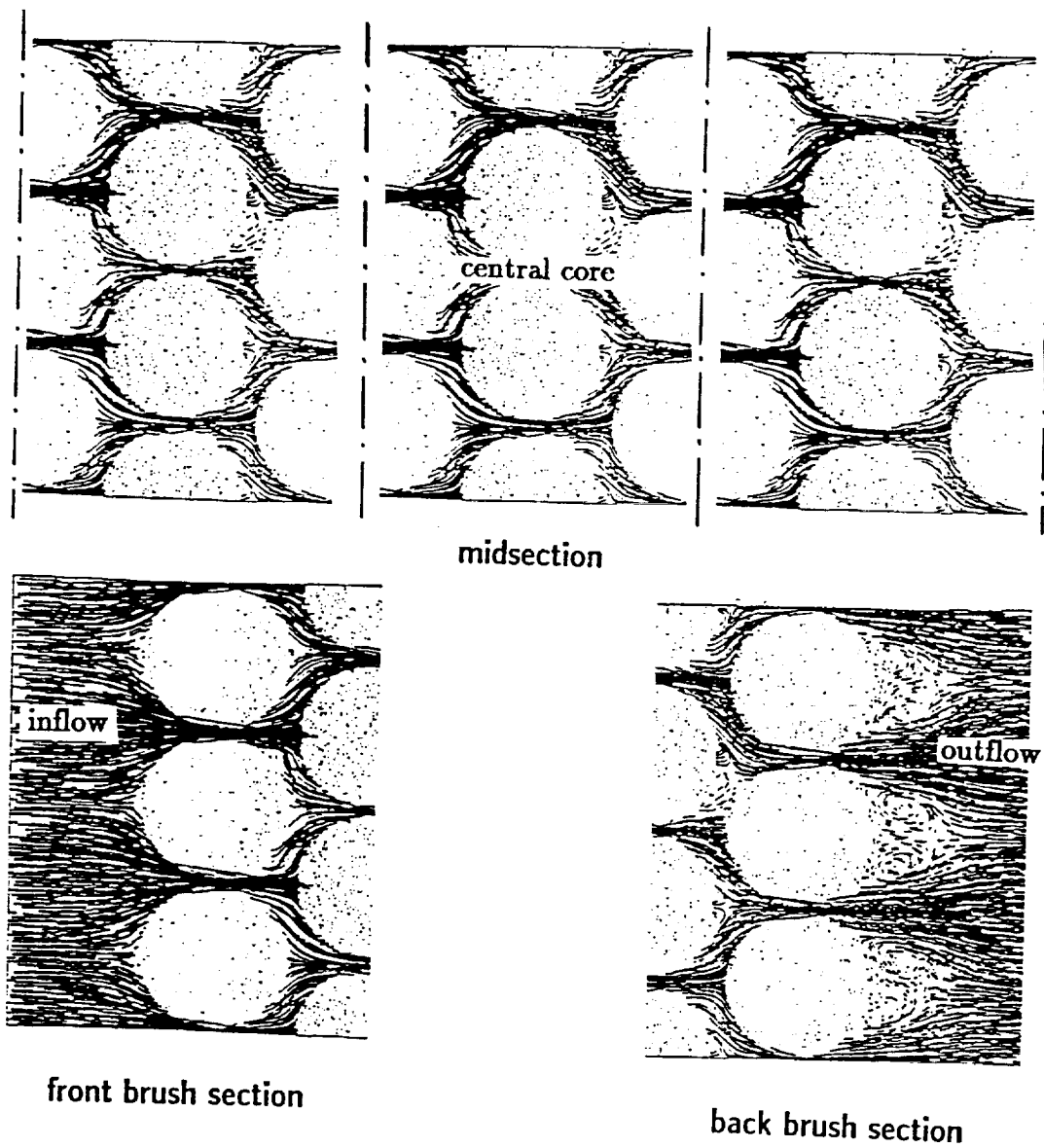
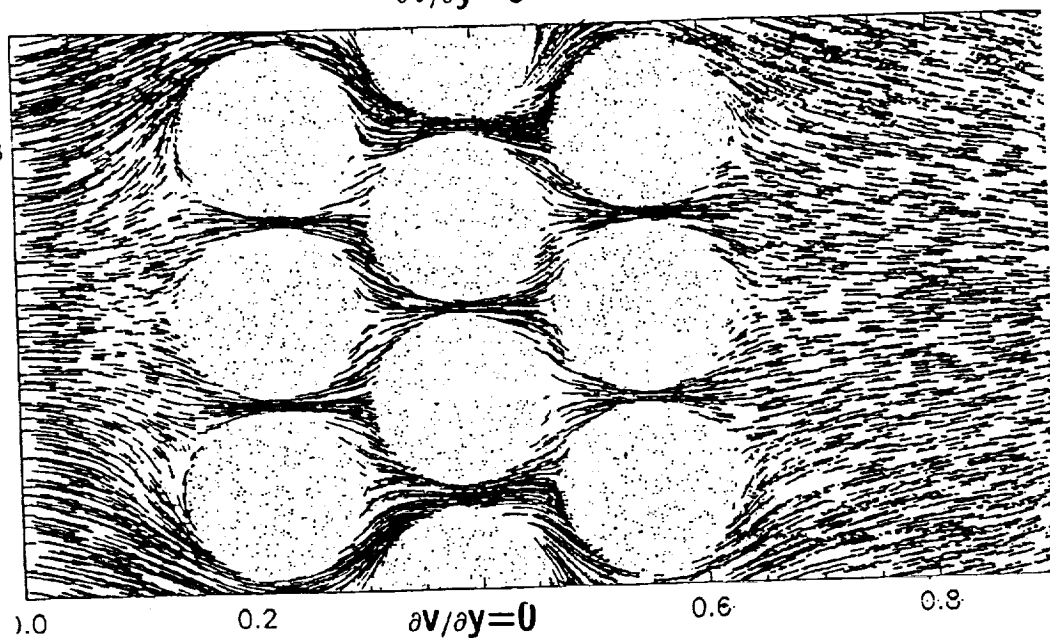


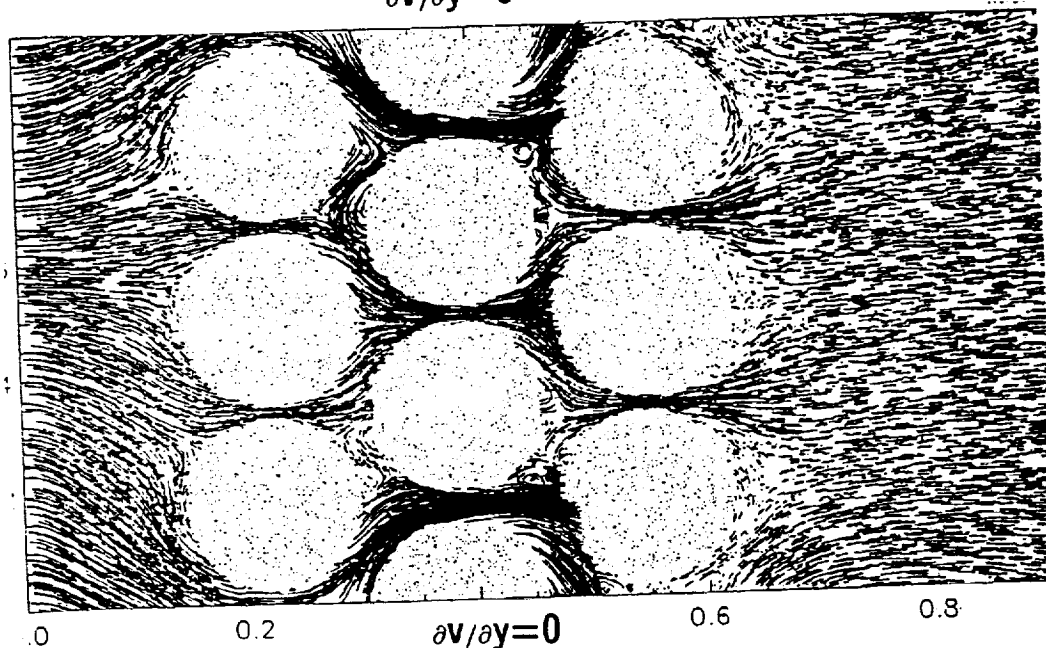
Figure 9. Assembled Brush Cluster at  $Re=20$

# STUDIES OF THE FLOW IN THE BRUSH SEGMENTS

Variation in the boundary conditions  
 $\partial v / \partial y = 0$



$\partial v / \partial y = 0$



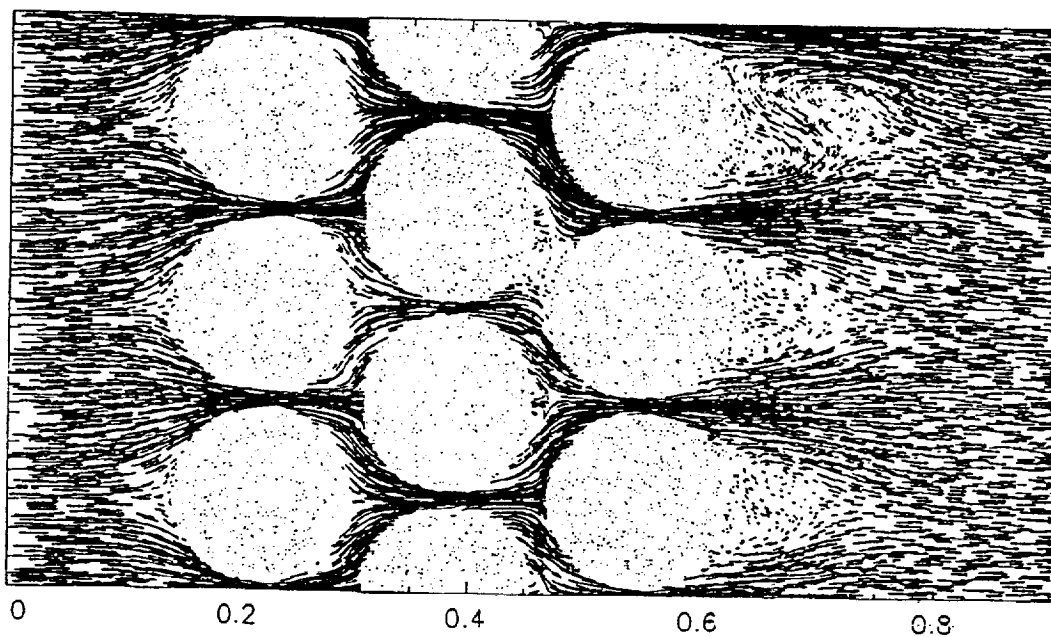
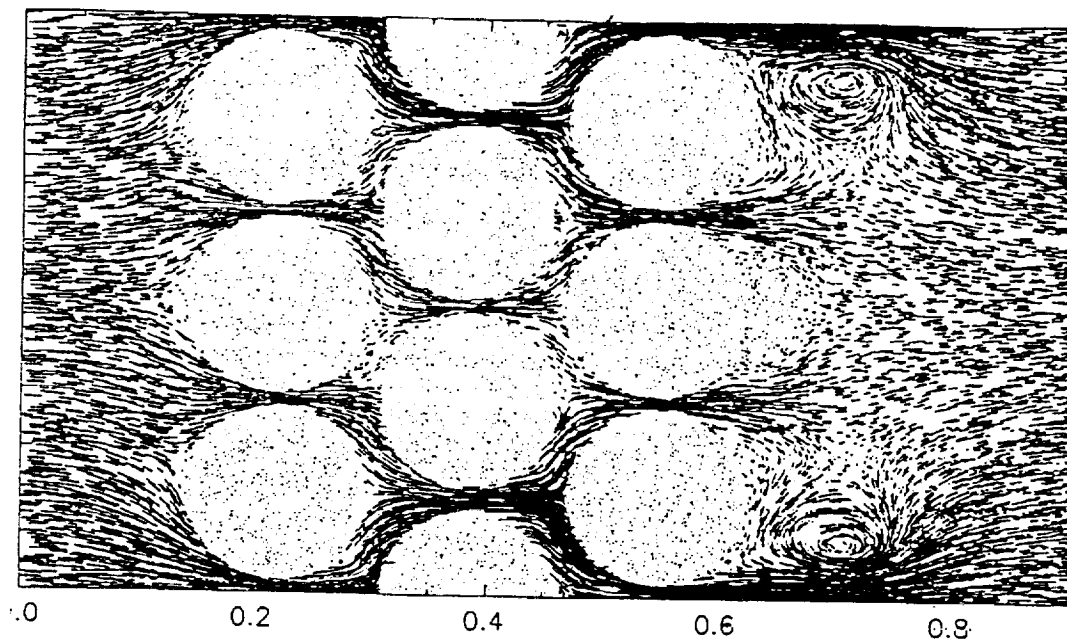
$Re_d = 20$

Boundaries are opened to the flow

## STUDIES OF THE FLOW IN THE BRUSH SEGMENTS

---

boundary pitch equal to the internal pitch



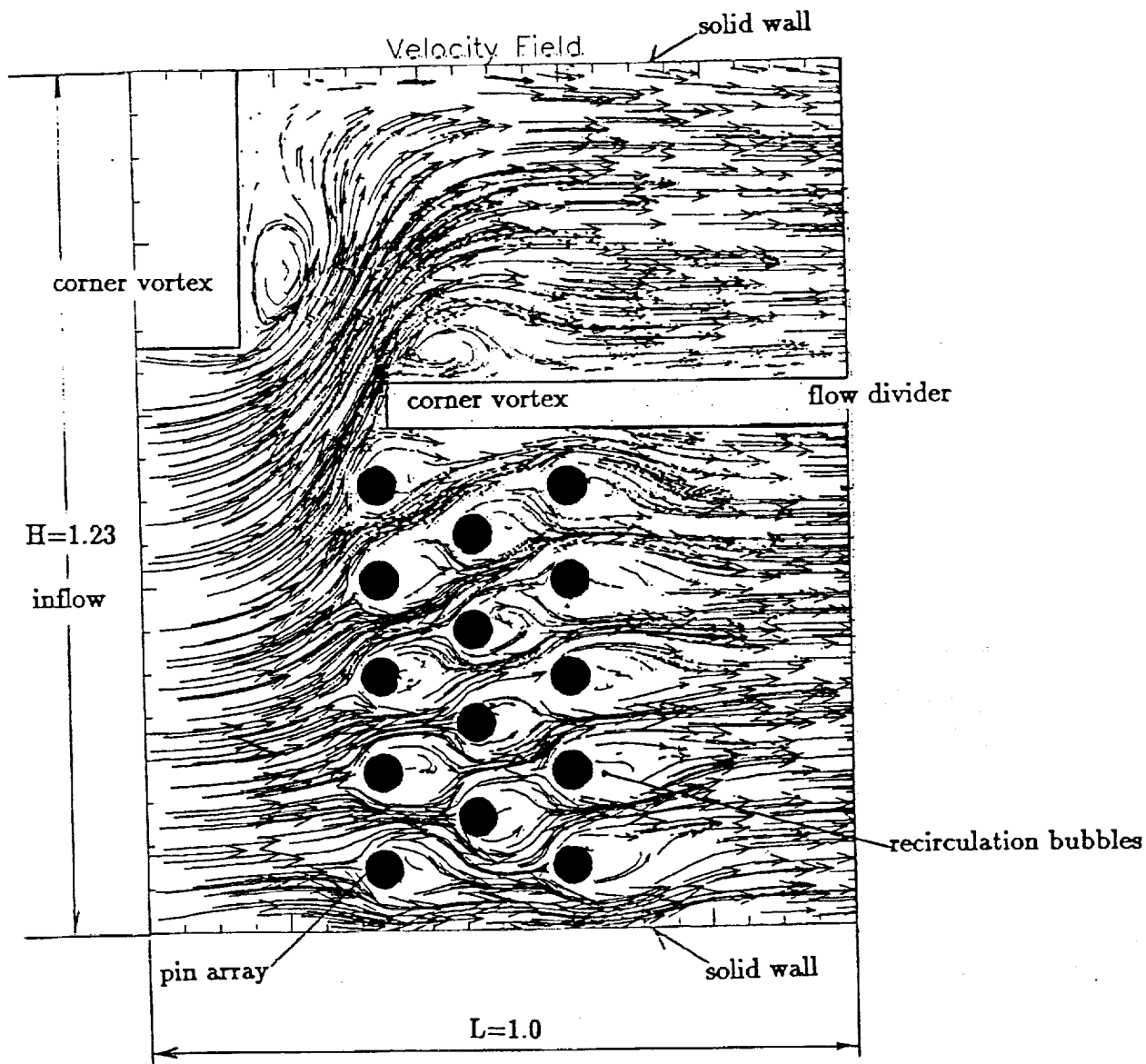
$t=1.4$

assymmetric configuration, top pitch larger than the bottom one

$Re_d=20$

## Cooling Pins : Stages of the Transient Flow Development

### OTHER POTENTIAL APPLICATIONS OF THE CODE:

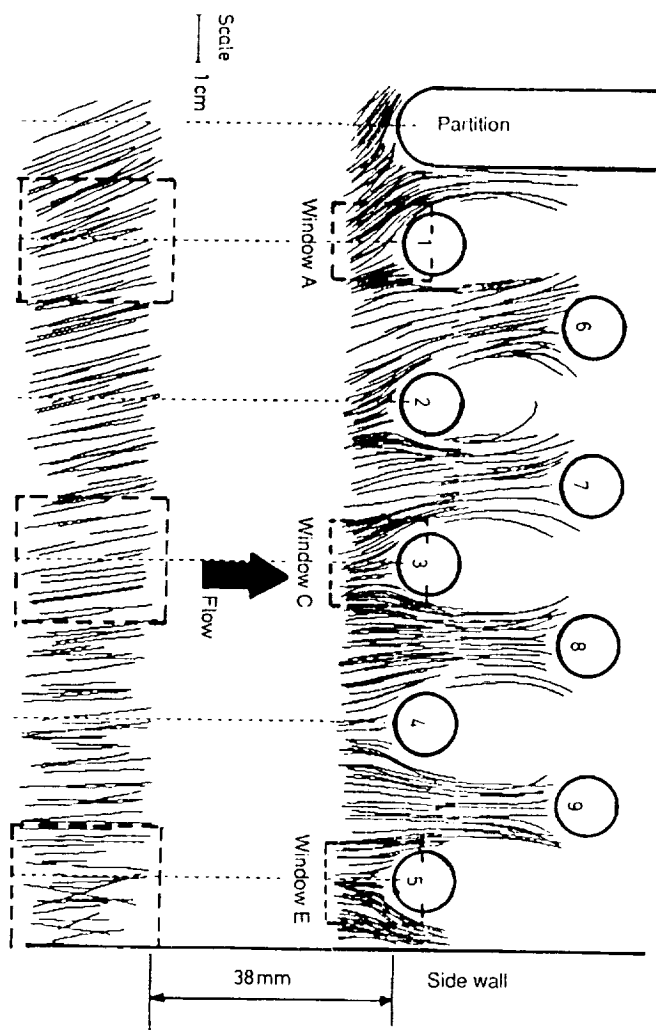


$$n_{it}=3000 \quad (\text{dimensionless time } T=h_0/U_0 \cdot t = 0.21)$$

$$Re_H=1000$$

$$U_0=0.5 \text{ m/s} \quad \rho_0=995 \quad (\text{working fluid - oil})$$





## CONCLUSIONS

---

The authors have presented chronologically the main contributors in the technological development of the brush seals. As one can see, there were three avenues that were followed. The first involved laboratory experiments on simulated brush seals, that led to a better understanding of the flow conditions and flow paths in the brush. The second involved industrial testing of brushes, that yielded real experimental data concerning pressure drops and leakages. These efforts performed within the framework of major industrial manufacturers and users, led to the incorporation of these seals in production jet engines. The third and final avenue was the numerical development of design and predictive tools for the brushes. These codes have used either lumped or distributed codes, as it was shown earlier in this paper. The present paper presents a first attempt at extending the distributed model of Braun and Kudriavtsev to a model where the brush is segmented in inlet, core and exit component. This concept allows the construction of infinitely large brushes with diminished computational penalty. It was shown that this concept predicts correctly the flow in the repetitive segments of the core, as well as in the inlet and exit zones.

## REFERENCES

---

1. Hendricks, R.C., Carlile, G.A., Liang A.D., "Some Sealing Concepts, Part B: Brush Seal Systems", Fourth International Symposium on Transport Phenomena and Dynamics of Rotating Machinery, Honolulu, Hawaii, ISROMAC-4, vol. A, April 5-8, 265, (1992).
2. Withers, P.A., "High Pressure Compressor Delivery Brush Seal of the International Aero Engines (IAE) V2500-A1 Gas Turbine Engine", Proceedings of Seals Flow Code Development-92, NASA CP-10124, pp. 275-279, NASA Lewis Research Center, Cleveland, Ohio August 5-6, 1992.
3. Shapiro, W., Artiles, A., "Industrial Code Development, Seals Flow Code Development-92", NASA CP-10124, Proceedings of a workshop held at NASA Lewis

- Research Center, August, 5-6, 1992, pp. 13-67.
4. Braun, M.J., Kudriavtsev, V.V., "A Numerical Simulation of a Brush Seal Section and Some Experimental Results", Int. Gas Turbine and AeroEngine Congress, Cincinnati, 1993, ASME Paper 93-GT-398, pp. 1-12. (Accepted for the publication in the ASME Journal of Turbomachinery).
  5. Liang A., (Editor), Seals Flow Code Development, NASA CP-10070, NASA Lewis Research Center, Cleveland, Ohio, March, 1991.
  6. Chupp, R.E., Holle, G.F., Dowler, C.A., "Simple Leakage Flow Model for Brush Seals", AIAA Paper 91-1913, AIAA/SAE/ASME/ASEE 27th Joint Propulsion Conference, June 24-26, Sacramento, CA, 1991.
  7. Hendricks, R.C., Shlumberger, S., Braun, M.J., Choy, F., Mullen, R.L., "A bulk flow model of a brush seal system", ASME Paper, 91-GT-325, 1991 .
  8. Flower, R. "Brush Seal Development Systems", AIAA Paper 90-2140, 1990.
  9. Holle, G., Krishnan, M., "Gas Turbine Engine Brush Seals Applications", AIAA Paper 90-2142, 1990.
  10. Chupp, R., Nelson, P., "Evaluation of Brush Seals for Limited Life Gas Turbine Engines", AIAA Paper 90-2140, 1990.
  11. Dowler, C., "Air Force Brush Seal Programs", Seals Flow Code Development-92, NASA CP-10124, Proceedings of a Workshop held at NASA Lewis Research Center, Aug 5-6, 1992.
  12. Basu, P.,
  13. Childs, D., "Dynamic Coefficients for Multiple Brush Seals", Seals Flow Code Development-92, NASA CP-10124, Proceedings of a workshop held at NASA Lewis Research Center, August, 5-6, 1992, pp. 211-213.
  14. Carroll, P.F., Easter B.P., "Brush Seal Would Impede Flow of Hot Gas", (Rockwell International), NASA Tech Briefs, Vol.17, No.12, Dec. 1993.
  15. Cannaci, V.A., Braun, M.J., Hendricks, R.C., "Flow Visualization and Motion Analysis for a Series of Four Sequential Brush Seals", J. of Propulsion and Power, Vol. 8, No. 3, pp. 697-702, 1992.
  16. Braun, M.J., Canacci, V.A., Russell, L.M., "Full field flow visualization and computer-aided velocity measurements in a bank of cylinders in a wind tunnel", Experiments in Fluids, vol. 13, pp. 117-127, 1992 .
  17. Zukauskas, A., "Heat Transfer from Tubes in Crossflow", Adv. Heat Transfer, Vol. 8, pp. 93-160, 1972.
  18. H.F., Griner, "Seal Related Activities at EG&G", NASA CP-10070, Proceedings of a Workshop held at NASA Lewis Research Center, March 26, 1991, pp.113-124 .

19. Kudriavtsev, V.V., "Numerical Studies of the Transient Flows within the Brush Seal Elements of the Aerospace Turbomachinery", Ph.D. Dissertation, Moscow Aviation Institute, School of Engine Technology, Oct., 1993(in Russian).
20. Kudriavtsev, V.V., Braun, M.J., "A Reynolds Number Parametric Numerical Investigation of Flow Structures and Pressure Distributions in a System of Cylinder Arrays", Int. ASME Fluid Engineering Conference, Washington D.C., Separated Flows/FED-Vol.149, Ed. by J.C.Dutton and L.P.Purtell, 1993, pp. 83-95 .
21. Braun, M.J., Kudriavtsev, V.V., "Numerical Visualization of Flow Structures in Dense Banks of Cylinders Located in a Channel", Third Int. Symposium on Experimental and Numerical Flow Visualization, 1993 ASME Winter Annual Meeting, New Orleans, Louisiana, Nov.28-Dec. 3, 1993/Eds. B. Khalighi, M.J. Braun, C.F. Freitas, D.H. Fruman, pp. 1-8. (submitted to the ASME Journal of Fluids Engineering)
22. Braun, M.J., Kudriavtsev, V.V., "Experimental and Analytical Investigation of Brush Seals", Seals Flow Code Development-92, NASA CP-10124, Proceedings of a workshop held at NASA Lewis Research Center, August, 5-6, 1992, pp. 181-195.
23. Ghia, K.N., Ghia, U., Oswald, G.A., "Study of Incompressible Separated Flow Using an Implicit Time-Dependent Technique", AIAA Paper 83-1894, 1, 1983.
24. Roache, P.L., Computational Fluid Mechanics, Hermosa Publishers, 1985.
25. Hendricks R.C., "Integrity Testing of Brush Seal in a T-700 Engine", Seals Flow Code Development-92, NASA CP-10124, Proceedings of a workshop held at NASA Lewis Research Center, August, 5-6, 1992, pp.117-138.
26. Gresho, P.M., Sani, R.L. Lee, R.L., Griffiths, D.F., "The Cause and Cure(?) of the Spurious Pressures Generated by Certain FEM Solutions of the Incompressible Navier-Stokes Equations", Int. J. Num. Meth. in Fluids, 1, No.1, 1981.
27. Patankar, S.V. Numerical Heat Transfer and Fluid Flow, Hemisphere, 1980.
28. Washpress, E.L., Iterative Solution of Elliptic Systems, Prentice-Hall, 1966.
29. Torrance, K.E., "Comparison of Finite-Difference Computations of Natural Convection", Journal Res. NBS: Math. Sci, N72B, 281, 1968.
30. Kudriavtsev, V.V., "New Higher-order Finite difference scheme for Hyperbolic Conservation Law: HYBRID Approach", Proc. Modeling and Simulation Conference, Eds. W.G.Vogt, M.H.Mickle, vol. 23, part 5, Pittsburgh, pp. 2655-2662, 1992.
31. Hayase, T., Humphrey, J.A.C., Grief, R., "A Consistently Formulated QUICK Scheme for Fast and Stable Convergence Using Finite-Volume Iterative Calculation Procedures", Journal of Computational Physics, Vol. 98, pp., 1992.
32. Zdravkovich, M.M., Stonebanks K.L., "Intrinsically Non-Uniform and Metastable Flow in and Behind Tube Arrays", Flow Induced Vibrations in Cylindrical Structures:

- Solitary Cylinders and Arrays in Cross-Flow, Vol. 1, pp. 61-73, 1988.
33. Mullen R., Braun, M., Hendricks R., "Numerical modeling in simulated brush seal configurations", AIAA Paper, AIAA-90-2141, 1990.
34. Launder, B.E., and Massey, T.H. "The Numerical Prediction of Viscous Flow and Heat Transfer in Tube Banks", Transactions of the ASME, Journal of Heat Transfer, Vol.100, pp.565-571, Nov. 1978 .

## **OBTAINED NEW SCIENTIFIC RESULTS :**

---

- ★ transients are negligible for  $Re < 1000$
- ★ pressure distribution and flow structure behind the simulated brush (located in the channel) are non-symmetrical. Flow asymmetry depends on the  $Re$  number.
- ★ creation of the butterfly-like flow formation behind the simulated brush seal (located in the channel) and observed experimentally and numerically
- ★ nonlinear behaviour of the pressure coefficient vs  $Re$  number for the brush segments located in the channel. Pressure paradox.
- ★ nonlinear behaviour of the pressure drop vs  $PTDR_L$
- ★ formation of the near wall jets for the multi-row cylinder bundles located within the channel (simulated brush), Coanda effect, flow "expansion" from the center to the walls

## CONCLUSIONS

---

- ★ approximate mathematical model with distributed parameters that is free from the disadvantages associated with porous media assumption
- ★ developed and evaluated a new computational algorithm for the solution of the N-S equations in  $(u,v,p)$  formulation
- ★ systematic analyses of the fluid flow in the brush seal components
- ★ Capabilities of calculation of the pressure drop for a given bristle geometry and the flow rate for typical brush segment

